ON THE TRANSITION BETWEEN RE-ENTRANT JET AND CONDENSATION SHOCK MECHANISM IN SHEET TO CLOUD CAVITATION

Diego Vaca-Revelo¹, Aswin Gnanaskandan¹,*

¹Mechanical and Materials Engineering Department, Worcester Polytechnic Institute, United States

ABSTRACT

Sheet to cloud cavitation over a wedge is investigated using numerical simulations performed at a Reynolds number \( Re = 200,000 \) and cavitation numbers ranging from \( \sigma = 1.44 \) to \( \sigma = 2.18 \). The multiphase fluid is described using a homogeneous mixture model, and the governing equations are the compressible Navier Stokes equations for the liquid/vapor mixture along with a transport equation for the vapor mass fraction. The numerical method, based on a characteristic based filtering approach for shock and interface capturing, is first validated by comparing with the experimental measurements, showing acceptable agreement. A systematic parametric investigation of sheet to cloud cavitation phenomenon for different cavitation numbers is then carried out. The two primary mechanisms known to destabilize the sheet cavity viz. the re-entrant jet mechanism and the condensation shock mechanism are captured in the simulations. Two cases in the condensation shock regime and two cases in the re-entrant jet regime are investigated in this paper; the similarities and differences between the two regimes are elucidated. The results presented here are part of an ongoing investigation on identifying the precise physical conditions that lead to the transition between the re-entrant jet and condensation shock mechanisms, a clear knowledge of which is yet to be elucidated.

Keywords: sheet to cloud cavitation, re-entering jet cycle, bubbly shock cycle

1. INTRODUCTION

Cavitation refers to the formation of vapor when the pressure in a liquid falls below vapor pressure. It occurs in a wide variety of situations ranging from orifices and valves to propulsor blades and turbomachinery. The formation of vapor is often followed by a growth of the vapor cavity and its violent collapse under a high-pressure environment. The physical consequences of this collapse include noise, vibration, surface erosion, and even compromised structural integrity. Sheet cavitation and its transition to cloud cavitation, a class of hydrodynamic cavitating flows, are of great practical interest since the highly unsteady nature of this flow can induce significant fluctuations in the thrust and torque of marine propulsors. Hence, a fundamental understanding of this phenomenon is necessary to mitigate and control its detrimental effects.

The transition of a sheet cavity to a cloud cavity is known to occur primarily through two mechanisms: liquid re-entrant jet and condensation shock. The liquid re-entrant jet mechanism has been investigated by various experimental and computational studies [1–4]. A sufficiently high adverse pressure gradient at the cavity closure is necessary for a re-entrant jet to develop, which travels upstream, splits the vapor cavity and results in shedding [5]. Different classes of re-entrant jet based on the thickness of the cavity have also been established [5]. Laberteaux and Ceccio [6] further classified cavities as open and closed based on the absence and presence of re-entering jet respectively. A closed cavity has a clear interface, and a re-entering jet is often found, whereas an open cavity is typically frothy with no clear re-entering jet.

At lower cavitation numbers, many authors [1, 7] have shown the presence of condensation shocks in a cavitating flow. The experiments of Ganesh et al [8] attributed the presence of a condensation shock as a mechanism for the transition from sheet to cloud cavitation. Budich et al [9] also demonstrated the presence of a condensation shock using their numerical model and showed that flow properties across these shocks obeyed the Rankine-Hugoniot jump conditions. Bhatt and Mahesh [10] simulated the same configuration as Ganesh et al [8] using Large Eddy Simulation (LES) to capture the condensation shock mechanism while pointing out differences in streamline curvature and pressure recovery between the re-entrant jet cases and condensation shock cases. According to them, a higher streamline curvature leads to a higher adverse pressure gradient and lower pressure recovery benefits shock wake propagation. Recently, Trummler et al. [11] investigated the dynamics of condensation shocks and re-entrant jets in a constant throat area nozzle using LES and suggested that...
re-entrant jet can transform into a condensation shock as it moves upstream.

Past studies on condensation shock [9–11] have simulated cases at only one cavitation number to demonstrate the presence of condensation shock. While they describe the formation and effects of condensations shock in detail, the fundamental knowledge behind how the mechanism switches from re-entrant jet to condensation shock is not clearly elucidated yet. Hence, this study aims to investigate the sheet to cloud cavitation over a wedge (corresponding to the experiments of Ganesh et al [8]) through a series of numerical simulations, focusing especially on the condensation shock mechanism and the physical conditions that lead to the transition between the re-entrant jet mechanism and condensation shock mechanism. Further, we intend to establish a series of cases that show condensation shock and study the similarity between these cases while contrasting them from the re-entrant jet cases.

While it is widely accepted these days that LES yields better predictions of sheet to cloud cavitation [12], we use the Unsteady Reynolds Averaged Navier Stokes equations (URANS) in this study, primarily to carry out a large number of parametric simulations which would not be feasible with LES. We expect that the mechanistic insights offered by URANS simulations will still be valuable and LES studies will be strategically used in the future to simulate a few cases from our parametric space to lend further credibility to our findings from URANS simulations. As preliminary results of our investigation, we report here results from four simulations (two in the condensation shock regime and two in the re-entrant jet regime) and an analysis of similarities and differences between them, with more simulations and extensive analysis planned for a future study. The paper is organized as follows. We briefly present the governing equations and numerical method used. Then the problem configuration, grid and boundary conditions are discussed. After a preliminary validation study of our model with experiments, the results from the four cases are analyzed and reported in detail.

2. GOVERNING EQUATIONS AND NUMERICAL METHOD

The multiphase medium is represented using a finite rate homogeneous mixture model [13] that assumes mechanical and thermal equilibrium between the liquid and the vapor phases. The governing equations are the compressible Navier Stokes equations for the liquid/vapor mixture along with a transport equation for the vapor mass fraction. The governing equations are:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} &= -\frac{\partial}{\partial x_k} (\rho u_k), \\
\frac{\partial u_k}{\partial t} &= -\frac{\partial}{\partial x_k} (\rho u_k u_i + p \delta_{ik} - \sigma_{ik}) , \\
\frac{\partial p e_v}{\partial t} &= -\frac{\partial}{\partial x_k} (\rho e_v u_k - Q_k) - p \frac{\partial u_k}{\partial x_k} + \sigma_{ik} \frac{\partial u_i}{\partial x_k}, \\
\frac{\partial \rho Y}{\partial t} &= -\frac{\partial}{\partial x_k} (\rho Y u_k) + S_e - S_c,
\end{align*}
\]

where \( \rho, u, e, \) and \( p \) are density, velocity, internal energy, and pressure of the mixture respectively and \( Y \) is the vapor mass fraction. The mixture density in a homogeneous mixture model is given by:

\[
\rho = \rho_l (1 - \alpha) + \rho_v \alpha,
\]

where \( \rho_l \) and \( \rho_v \) are the density of the liquid and vapor, respectively. \( \alpha \) is the vapor volume fraction and is related to the vapor mass fraction by,

\[
\rho_l (1 - \alpha) = \rho (1 - Y) \quad \text{and} \quad \rho_v \alpha = \rho Y.
\]

The mixture equation of state is given by [14]

\[
p = Y \rho R_g T + (1 - Y) \rho K_l T - \frac{p}{p + P_c},
\]

where \( R_g = 461.6 \) J/KgK, \( K_l = 2684.075 \) J/KgK, and \( P_c = 786.33 \) MPa are constants associated with the equation of state of vapor and liquid. The equation to evaluate internal energy is given by

\[
\rho e_v = \rho C_{em} T + \rho (1 - Y) \frac{P_c K_l T}{p + P_c},
\]

\[
C_{um} = (1 - Y) C_{vl} + Y C_{vg}.
\]

\( C_{vl} \) and \( C_{vg} \) are the specific heats at constant volume for liquid and vapor respectively. The viscous stress \( \sigma_{ij} \) and heat flux \( Q_l \) are

\[
\sigma_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right), \quad Q_l = k \frac{\partial T}{\partial x_i},
\]

where \( \mu \) is the mixture viscosity and \( k \) is the mixture thermal conductivity. The source terms for evaporation of liquid \( S_e \) and condensation of vapor \( S_c \) are given by [15]

\[
S_e = C_e \alpha^2 (1 - \alpha)^2 \frac{D_l \max(p_v - p, 0)}{\rho_g \sqrt{2 \pi R_g T}}, \quad S_c = C_e \alpha^2 (1 - \alpha)^2 \frac{\max(p - p_w, 0)}{\sqrt{2 \pi R_g T}},
\]

where \( p_v \) is the vapor pressure. \( C_e \) and \( C_c \) are empirical constants whose value is 0.1. The vapor pressure can be calculated by

\[
p_v = p_k \exp \left[ \left( 1 - \frac{T_k}{T} \right) \left( a + (b - cT)(T - d)^2 \right) \right],
\]

\[
p_k = 22.130MPa, \quad T_k = 647.31K, \quad a = 7.21, \quad b = 1.152 \times 10^{-5}, \quad c = -4.787 \times 10^{-9}, \quad d = 483.16.
\]

The simulations use the algorithm developed by Gnanaskandan and Mahesh [13] for cavitating flows on unstructured grids. The algorithm makes use of a novel predictor corrector approach to handle shocks and material interfaces in the flow with minimal dissipation. The algorithm has been validated for a variety of flows including a turbulent cavitating flow over a hydrofoil [13, 16], and a hemispherical headform [17]. In the current study, an unsteady RANS method has been used since it has the
advantage of being computationally less expensive than LES. We use the Spalart-Allmaras [18] model given by

\[
\frac{\partial \rho \tilde{v}}{\partial t} + \frac{\partial (\rho \tilde{v} u_k)}{\partial x_k} = c_{b1} \rho \tilde{v} + \frac{1}{\sigma} [(1 + c_{b2}) \nabla \cdot ((\rho \tilde{v} + \rho \tilde{v}) \nabla \tilde{v}) - c_{b2} (\rho \tilde{v} + \rho \tilde{v}) \nabla \cdot \nabla \tilde{v}] - \rho c_{w1} f_w \left( \frac{\tilde{v}^2}{d^2} \right),
\]

(9)

where \( \nu_T = \tilde{v} f_{e1}, f_{e1} = x^3/(x^3 + c_{v1}) \) and \( \chi = \tilde{v}/\nu \). \( S \) is the strain rate. The model is closed with the following coefficients and wall functions:

\[
\tilde{S} = S + \frac{\tilde{v}}{k^2 d^2} f_{e2}, \quad f_{e2} = \left( 1 + \frac{\chi}{c_{v2}} \right)^{-3},
\]

\[
f_w = g \left( \frac{1 + c_{w3}}{g^6 + c_{w4}} \right)^{1/6}, \quad g = r + c_{w4}(r^6 - r),
\]

\[
r = \frac{\tilde{v}}{k^2 d^2}, \quad \sigma = \frac{2}{3}, \quad c_{b1} = 0.1355,
\]

(10)

\[
c_{b2} = 0.622, \quad c_{v1} = \frac{c_{b1}}{k^2} + \frac{1 + c_{n2}}{\sigma}, \quad c_{w3} = 0.3,
\]

\[
c_{u3} = 2, \quad c_{v1} = 7.1, \quad c_{v2} = 5, \quad \kappa = 0.41.
\]

Additionally, it uses the eddy viscosity modification suggested by Coutier-Delgosha et al [19]. They observed that the eddy viscosity evaluated from standard RANS models can be excessive, which prevents cloud formation. Hence, the modified eddy viscosity is given by

\[
\mu_T = \nu_T \left[ \rho_0 + (\rho_1 - \rho_0)(1 - \alpha)^1 \right].
\]

(11)

The turbulent thermal conductivity and turbulent scalar diffusivity are computed assuming a turbulent Prandtl number \( (Pr_t) \) of 0.9 and a turbulent Schmidt number \( (Sc_t) \) of 0.7. The turbulent scalar equation is then modified as

\[
\frac{\partial \rho Y}{\partial t} = - \frac{\partial}{\partial x_k} (\rho Y u_k) + S_c - S_e + \frac{\partial}{\partial x_k} \left( \frac{v_i}{Sc_t} \frac{\partial \rho Y}{\partial x_k} \right).
\]

(12)

3. PROBLEM DESCRIPTION

Figure 1 shows a schematic of the problem which is based on the experiments of Ganesh et al [8]. Even though the experimental set up is a three dimensional geometry [8], in this work, the transition from sheet to cloud cavitation is simulated in a two dimensional domain. This assumption is justified since we use the unsteady RANS method in which the averaged variations in the spanwise direction are considered to be much smaller compared to the averaged changes in the streamwise and wall-normal directions. Hence, the major changes in the fluid properties are anticipated to occur in the streamwise and wall-normal directions.

The wedge apex is located at the origin, and the geometric parameters of the computational domain, as a function of the height of the wedge \( (h = 1 \text{ inch}) \), are shown in the Figure 1. The domain is extended upstream and downstream to minimize the effect of acoustic reflection from the boundaries. Additionally, acoustically absorbing sponge layers [13] (red rectangular regions in Figure 1) are applied at the inlet and outlet to prevent acoustic reflections. Uniform inflow velocity is prescribed at the inlet boundary (left boundary). At the outlet (right boundary), the gradient of all the variables is equalized to zero except for pressure, and the top and bottom boundaries are considered as walls with a no-slip boundary condition. The Reynolds number of the flow based on the wedge height and free stream velocity \((7.9 \text{ m/s}) \) is 200, 000.

It is expected that the major vapor production takes place along the length of the wedge, so the grid was made very fine in this region. The minimum grid spacing near the wedge is 0.001 \( h \times 0.001 h \) in the normal and streamwise directions, respectively. The wall-normal spacing stretches to 0.005 \( h \) at a height of 0.5 \( h \) from the wedge apex and further to about 0.01 \( h \) at a height of \( h \) from the apex. In the streamwise direction, the grid is stretched to 0.02 \( h \) at 3.5 \( h \) from the apex and further to 0.01 \( h \) at the end of the wedge.

A justifiable starting point for this study is to reach one of the conditions measured experimentally and then carry out a systematic parametric investigation at several cavitation numbers. In the experimental study, velocity measurements are taken from a plane (E1) located at 3.25 \( h \) upstream of the wedge apex, and pressure measurements from another plane (E2) at 22 \( h \) downstream of wedge apex, to compute the operating cavitation number. In the simulations, the conditions at the computational boundaries are iteratively changed to match with the experimental values at E1 and E2, so that the same operating cavitation number as in the experiments is attained. The experimental velocity at E1 is achieved by adjusting the inlet velocity, and the measured pressure at E2 is reached by calibrating the pressure at the outflow boundary.

4. RESULTS AND DISCUSSION

Four different cases (two in the condensation shock regime and two in the re-entrant jet regimes) at different cavitation numbers have been carried out. Table 1 shows the parameters that characterize each case. \( L_c \) is the time-averaged length of the cavity and \( t_c \) is its time-averaged maximum thickness. The time-

<table>
<thead>
<tr>
<th>Study case</th>
<th>Inlet velocity</th>
<th>Cavitation number</th>
<th>( L_c/h )</th>
<th>( t_c/h )</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>7.98 m/s</td>
<td>2.18</td>
<td>1.91</td>
<td>0.30</td>
</tr>
<tr>
<td>B</td>
<td>7.97 m/s</td>
<td>2.08</td>
<td>2.39</td>
<td>0.34</td>
</tr>
<tr>
<td>C</td>
<td>7.91 m/s</td>
<td>1.62</td>
<td>5.39</td>
<td>0.68</td>
</tr>
<tr>
<td>D</td>
<td>7.89 m/s</td>
<td>1.44</td>
<td>8.23</td>
<td>0.99</td>
</tr>
</tbody>
</table>
averaged fluid properties of cases A, B, C, and D were computed using data sampled each 0.005 $t_u/\delta$ of thirteen, seven, six, and four shedding cycles, respectively. The inlet velocities are overall maintained constant in the four study cases, and the cavitation numbers ($\sigma$) vary ranging from 1.44 to 2.18. In general terms, the largest and thickest partial cavity was attained at the lowest cavitation number. Figure 2 shows a schematic of the time averaged cavity where the time averaged cavity length $L_c$, time averaged cavity thickness $t_c$, wedge parallel and perpendicular axis ($s$-axis, and $n$-axis) are shown. Note that $L_c$ is measured along the $s$-axis and $t_c$ along the $n$-axis.

4.1 Comparison to experiment

The numerical approach is first validated at a cavitation number $\sigma = 1.62$, which corresponds to case C, by comparison with the experimental measurements at $\sigma_{exp} = 1.73 \pm 0.11$ reported by Ganesh et al \[8\]. The simulated time-averaged values of void fraction shown in Figure 3 correspond to six shedding cycles and have approximately 20,100 samples. The values of mean void fraction predicted by the simulation at streamwise stations $x/\delta = 0.75, x/\delta = 1.4$ and $x/\delta = 2.8$ show a close agreement with the experimental values thus providing confidence in the capability of the URANS simulations. These values also agree well with the URANS simulations conducted in the past by Gnanaskandan and Mahesh \[4\] using the same physical model and numerical method used in this study. The two dimensional assumption of the computational domain induces an error in the validation because the experiment considers the effects of the streamwise vorticity due to the three dimensional nature of the experimental set up; however, the streamwise vorticity is not directly resolved in our two dimensional simulations. This error is not dominant because the number of shedding cycles used to calculate the simulated time-averaged void fraction is acceptable enough to capture the unsteadiness of the flow.

FIGURE 2: SCHEMATIC OF A PARTIAL CAVITY AND ITS CHARACTERISTIC DIMENSIONS AT THE CRITICAL LENGTH.

FIGURE 3: MEAN VOID FRACTION PROFILES COMPARISON AT DIFFERENT STREAMWISE LOCATIONS. SYMBOLS: EXPERIMENT, LINES: NUMERICAL.

FIGURE 4: VOID FRACTION CONTOURS IN STUDY CASE A SHOWING THE TIME EVOLUTION OF CAVITY DESTABILIZATION AND SHEDDING DUE TO THE RE-ENTRANT JET MECHANISM AT $t_u/\delta = 116.0$ (a), 116.2 (b), 116.4 (c), 116.8 (d), 117.0 (e), AND 117.2 (f).
4.2 Time evolution of the cavity

The preliminary numerical results confirm that the attached sheet cavity forms at the apex wedge and grows up to a critical length, after which it sheds into a cloud cavity. The simulations also predict both re-entrant jet and condensation shocks to be the physical mechanisms of the transition. The cases A and B belong to the re-entrant jet regime and cases C and D belong to the condensations shock regime.

Figure 4 shows the time evolution of cavity destabilization and shedding due to a re-entrant jet starting when a critical length is reached (see Figure 4a). Figure 4b shows a thin liquid layer between the wall of the wedge and the cavity closure. This re-entrant jet then moves upstream and cuts the vapor pocket (see Figure 4b-4e). Finally, a vapor cloud is shed and convected downstream (see Figure 4f). A series of snapshots that capture the vapor shedding due to condensation shock is shown in Figure 5. In the first snapshot, the instantaneous void fraction values show high levels of vapor across the entire pocket, which follows the observations of Ganesh et al [8]. Then, the condensation front moves upstream until it reaches the wedge apex (see Figure 5b-5e) and finally a vapor cloud is shed that is convected downstream (see Figure 5f). Bhatt and Mahesh [10] suggested that the origin of this shock front is related to the pressure waves generated by the collapse of vapor clouds downstream which is also corroborated from our simulations. Note that as the shock front moves upstream, the void fraction ahead of the front is higher than the values behind it because it condenses the vapor as it moves through the vapor cloud.

4.3 Systematic comparison between the condensation shock and re-entrant jet cases

A comparison of tangential velocity along profiles normal and parallel to the wedge are shown in Figure 6 and Figure 7. Three streamwise stations are chosen for the wall normal profiles and four wall-normal stations are chosen for the wall parallel profiles. The presence of a re-entrant jet is evident at the station \( s/L_c = 0.7 \) for cases A and B in the form of a negative velocity near wall. While reverse flow is present even for cases C and D, it will be clear from the void fraction profiles in Figure 8 that this reverse flow is not a liquid re-entrant jet. From the profiles at \( s/L_c = 0.3 \) and 0.1, it is also evident that the free stream velocity is attained for cases A and B at a much closer distance from wall, indicating the thickness of the vapor cavity. In the wall parallel profiles at \( n/t_c = 0.05 \) and \( n/t_c = 0.1 \), the tangential velocity of cases A and B also show negative values corroborating the
presence of a re-entering jet especially when approaching the cavity closure. Further, as we move away from the wall (at stations \(n/t_c = 0.3\) and \(n/t_c = 0.6\), the negative velocity is no longer present for cases A and B, since the re-entrant jet is a thin liquid jet that is present only close to the wall. Cases C and D at the stations close to the wall do not show significant negative velocities since the primary mechanism of cavity destabilization is due to the condensation shock.

Time-averaged void fraction profiles are shown along wall normal and wall parallel lines in Figure 8 and Figure 9. In Figure 8, Case D shows higher amounts of vapor compared to the others, and case A is the one with the least vapor generation according to their respective cavitation numbers. It can also be observed from stations \(s/L_c = 0.3\) and 0.7 that there is no significant reduction in the vapor content close to the wall which would be the case had a re-entrant jet been present as in cases A and B. The thicker cavities for cases C and D are also evident from these plots. Figure 9 shows that for cases C and D, the mean void fraction is higher in regions closer to the wedge apex. Further, the slope of the void fraction reduction is also considerably high for the condensation shock cases, which could be an indicator of the stronger condensation effect of the condensation shock. However, this is only a hypothesis at this point, since further analysis on condensation rate due to a condensation shock is needed to confirm this, which will be a point of investigation in our future studies. This behavior though is limited only to the profiles closer to the wall and the profiles are similar to each other away from the wall for both the re-entrant jet and condensation shock cases.

**Figure 7:** Mean tangential velocity variation at different stations parallel to the wedge.

**Figure 8:** Mean void fraction variation at different stations normal to the wedge.

**Figure 9:** Mean void fraction variation at different stations parallel to the wedge.

**Figure 10:** Mean pressure variation at different stations parallel to the wedge.
Mean pressure profiles parallel to the cavity are shown in Figure 10. An adverse pressure gradient ($\partial P/\partial s > 0$) is present in all the cases, but a sharp pressure rise differentiates cases A and B from the others. In fact, this difference in the magnitude of adverse pressure gradient is present in stations both close to and away from the wall. Hence, it confirms that there is an adverse pressure gradient threshold that must be overcome to develop a re-entrant jet, which agrees with Bhatt and Mahesh [10]. Our future investigations will involve more re-entrant jet and condensation shock cases to confirm if the findings from Figure 10 are indeed universal in nature and that the curves from different re-entrant jets will collapse on each other as do the different condensation shock cases.

4.4 Rankine-Hugoniot jump conditions

The Rankine-Hugoniot jump conditions describe how fluid properties vary due to the passage of a shock wave. We use this approach to verify whether the observed phenomenon represents a condensation shock. The Rankine-Hugoniot jump conditions in a frame of reference moving with the shock are given by [20]

$$\rho_L \hat{u}_L^2 + p_L = \rho_R \hat{u}_R^2 + p_R, \quad \rho_L \hat{u}_L \left( e_L + \frac{p_L}{\rho_L} + \frac{\hat{u}_L^2}{2} \right) = \rho_R \hat{u}_R \left( e_R + \frac{p_R}{\rho_R} + \frac{\hat{u}_R^2}{2} \right),$$

(13)

where $\hat{u} = u - S$. The subscript L and R denote the left and right flow properties across the shock. The approach assumes a planar one-dimensional shock propagating within a homogeneous medium. The shock speed can be evaluated by

$$S = u_L - \sqrt{\frac{(\rho_R - \rho_L) \frac{\gamma_R}{\gamma_L} + 1}{(\rho_R - \rho_L) \frac{\gamma_L}{\gamma_R} + 1}},$$

(14)

where

$$\frac{1}{\gamma - 1} = \frac{C_{pm} p + [C_{pm} + (1 - \gamma) K_1]}{Y R_g (p + P_c) + K_1 (1 - \gamma) p},$$

(15)

The Mach number ($Ma$) of the shock wave is given by

$$Ma = \frac{S}{a},$$

(16)

where $a$ is the speed of sound, which is given by

$$a^2 = \frac{C_1 T}{C_0 - \frac{C_1}{C_{pm}}},$$

$$C_0 = 1 - (1 - Y) \rho K_1 T \left( \frac{P_c}{p + P_c} \right)^2, \quad C_1 = \frac{R_g Y + K_1 (1 - Y)}{p + P_c},$$

$$C_{pm} = Y C_{pg} + (1 - Y) C_{pl},$$

where $C_{pl}$ and $C_{pg}$ are the specific heats at constant pressure for liquid and vapor respectively.

Properties across the condensation front are extracted for cases C and D. In Tables 2 and 3, properties ahead (L) and behind (R) the front, speed of sound ($a$), shock speed ($S$), and Mach number ($Ma$) of the shock wave are shown. The properties presented in Tables 2 and 3 satisfy the Rankin-Hugoniot conditions with errors of 0.6% and 2.14%, respectively. The supersonic nature of the condensation fronts reaffirms that they are indeed shock waves propagating within the mixture.

5. CONCLUSION

The paper presents preliminary results from an ongoing investigation aimed at elucidating the fundamentals of the transition of sheet cavitation to cloud cavitation due to the condensation shock mechanism. Towards this, we present sheet to cloud cavitation studies due to condensation shock at two different cavitation numbers and compare and contrast these flows with a cavitating flow where the transition happens due to the well-known re-entrant jet mechanism. When normalized appropriately the two condensation shock cases seem to display similarity in certain parameters and show significant differences from the reentrant jet cases. The Rankine-Hugoniot jump conditions across the condensation shock reveal a supersonic shock front moving upstream in the vapor cavity. Future investigations will focus on simulating more condensation shock and reentrant jet cases and further analysis to identify the precise physical conditions that lead to the switching between these two mechanisms.

ACKNOWLEDGMENTS

We gratefully acknowledge the Fulbright Scholarship awarded to DVR and the start-up funds provided by Worcester Polytechnic Institute (WPI) to AG. Computing time provided
REFERENCES


